INVESTIGATION OF UNIFORMITY AND PRESSURE RECOVERY IN TURNING DIFFUSER BY MEANS OF BAFFLES

NUR HAZIRAH BINTI NOH @ SETH

A project report submitted in partial fulfillment
of the requirement for the award of the
Degree of Master of Mechanical Engineering

Faculty of Mechanical and Manufacturing Engineering
University Tun Hussein Onn Malaysia

DECEMBER 2012

ABSTRACT

Turning diffuser is an engineering device that is widely used in the industry to reduce the flow velocity as well as changing the direction of the flow. Having a curvature shape causes its performance to decrease in terms of pressure recovery (C_p) and flow uniformity (σ_u). Therefore, this study presents the works in designing baffle to be installed into the turning diffuser with area ratio of AR=2.16 to improve the flow uniformity and pressure recovery. It also aims to investigate the mechanism of flow structure and pressure recovery in turning diffusers by means of turning baffles. The results with varying inflow Reynolds number (Re_{in}) between 5.786E+04 – 1.775E+05 have been experimentally tested and compared with previous study. Particle image velocimetry (PIV) was used to determine the flow uniformity. On the other hand, digital manometer provides the average static pressure of the inlet and outlet of turning diffuser. The best produced pressure recovery of C_p=0.526 were recorded when the system were operated at the highest Reynolds number tested Re_{in}=1.775E+05. This result shows an improvement up to 54.625% deviation from previous study with C_p=0.239. The flow uniformity also shows an improvement of 47.127% deviation from previous study at the same Re_{in} with σ_u=3.235 as compared to previous study σ_u=6.12. However, the flow structure between baffles is not quite visible due to limitations of fog generator and camera position. Upon improvement of flow uniformity and pressure recovery, it is believed that turning diffuser performance could be improved with installing baffles.

ABSTRAK

Alat kejuruteraan yang selalu digunakan di dalam industri untuk mengurangkan halaju, dalam masa yang sama mengubah arah aliran ialah peresap membelok. Oleh kerana kriteria peresap membelok mempunyai bentuk lengkungan yang tidak dapat dielakkan, prestasinya akan terganggu dalam bentuk pemulihan tekanan *(C_p)* dan juga keseragaman aliran *(σ_u)*. Oleh yang sedemikian, penyelidikan ini membentangkan hasil kerja mengubahsuai bilah yang akan dipasang di dalam peresap membelok untuk meningkatkan pemulihan tekanan dan juga keseragaman aliran. Penyelidikan ini juga mempunyai objektif untuk mengkaji struktur aliran di dalam peresap membelok apabila bilah dipasang. Hasil dari penyelidikan ini, dengan nombor Reynolds *(Re_{in})* diubah suai dalam jurang 5.786E+04 – 1.755E+05 yang dijalankan dalam eksperimen, dibandingkan dengan hasil penyelidikan yang telah dikecapi sebelum ini. *Particle image velocimetry (PIV)* telah digunakan untuk menentukan keseragaman aliran. Dalam masa yang sama, manometer digital dapat mengukur purata tekanan statik di permukaan masuk dan keluar peresap membelok. Eksperimen ini didapati dapat menghasilkan pemulihan tekanan C_p=0.526 apabila dijalankan pada nombor Reynolds yang tertinggi iaitu Re_{in}=1.775E+05. Hasil ini menunjukkan peningkatan sebanyak 54.625% berbanding hasil penyelidikan sebelum ini dengan C_p=0.239 pada nombor Reynolds yang sama. keseragaman aliran juga menunjukkan peningkatan sebanyak 47.127% pada nilai nombor Reynolds yang sama dengan σ_u=3.235 jika dibandingkan dengan penyelidikan sebelumnya iaitu σ_u=6.12. Walau bagaimanapun, bentuk aliran di antara bilah tak dapat dilihat dengan jelas kerana faktor asap putih dan kedudukan kamera yang tidak dapat diubahsuai lagi. Kesimpulannya, dalam memperbaiki keseragaman aliran pemulihan tekanan, prestasi peresap membelok dapat diperbaiki dengan memasang bilah di dalamnya.

CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS AND ABBREVIATIONS

c	- Distance between leading edge to trailing edge of an airfoil
C_d	- Friction coefficient
C_p	- Pressure recovery coefficient
D	- Diameter
d	- Spacing between guide vanes (airfoil)
e	- Expansion ratio
K	- Loss coefficient
L_{in}	- Length of inner wall of diffuser
P	- Pressure
r	- Radius
Re_{in}	- Reynolds number
u	- Velocity in X-direction
v	- Velocity in Y-direction
w	- Velocity in Z-direction
V	- Velocity
W	- Height if inlet diffuser
σ_u	- Standard deviation
θ	- Angle
π	- PI number
ρ	- Density
μ	- Dynamic viscosity
ATP	- Aeronautic Test Program
CFD	- Computational Fluid Dynamic
NASA	- National Aeronautics and Space Administration
UTHM	- University Tun Hussein Onn Malaysia

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

Diffuser is a common engineering device which has the simplest design of an expanding area in the flow direction. The main and basic function of a diffuser is to decelerate the velocity of the flow. In contrast, a nozzle is often used to accelerate the flow and direct the flow in one particular direction. A diffuser increases the pressure of the fluid by slowing it down. The cross section area of a diffuser increases in the flow direction for subsonic flow and decreases for supersonic flows.

A wind tunnel is one of the many applications that utilize diffusers. Basically, a wind tunnel is a closed tubular passage with the object under test is mounted in the middle. A wind tunnel is constructed to test the aerodynamic of many objects. It is widely use by engineers to study the flow of air or gases around an object in motion, ranging from as small as tennis ball to as large as the entire airplane. From these studies, a feasible design with the best performance of a certain vehicles can be evaluated.

The airflow may have smoke or dye injected to make the flow line visible around the object under test. Aircraft engineers use wind tunnel to improve airplanes and spacecraft design and performance. In aircraft design, the pressure distribution is primarily responsible for lift and contributes to the drag of the aircraft. Automobile makers also rely heavily on wind tunnels. Race-car makers particularly use wind tunnels to improve their car's aerodynamics, speed and efficiency to get an advantage in the race.

A relatively small wind tunnel can only used to test a scaled-down model of an object, such as an entire airplane in miniature size, whereas others are large enough to test full-sized vehicles. The world largest wind tunnel is located at National Aeronautics and Space Administration (NASA) Ames Research Center located near San Jose, California. This particular large wind tunnel is about 54.8 meters high, more than 426.7 meters long and the test section can accommodate a plane with 30m wingspan.

Wind tunnels are usually designed for specific purpose and speed range. NASA has invented special program call Aeronautic Test Program (ATP). This program were designed ensure that wind tunnels meet the test the facilities for flight operations and available for United States government, corporation and institutions. This program provides facilities for specific function which consists of subsonic facilities, transonic, supersonic, hypersonic, propulsion and flight research facilities.

Referring to Figure 1.1, there are 5 major sections in a closed circuit wind tunnel: the settling chamber, contraction cone (nozzle), test section, diffuser and drive section. Typically air is moved through the tunnel using a series of fans. The air moving through the tunnel needs to be relatively laminar. When air started to enter the tunnels, the turbulence flow will settled and straightened by settling chamber with the help of steady flow engineering device such as honeycomb hole or mesh screen.

To increase the air velocity, it is forced into the contraction cone and passes through the test section where the sample is located. High velocity air flows over the sample and aerodynamic structure and result can be seen through the control panel screen. The air will subsequently flow into the diffuser which slows the velocity of the flow. When arrive at the drive section, the fan will creates high-speed flow and pull air into a smooth stream back to the settling chamber and repeats the whole process.

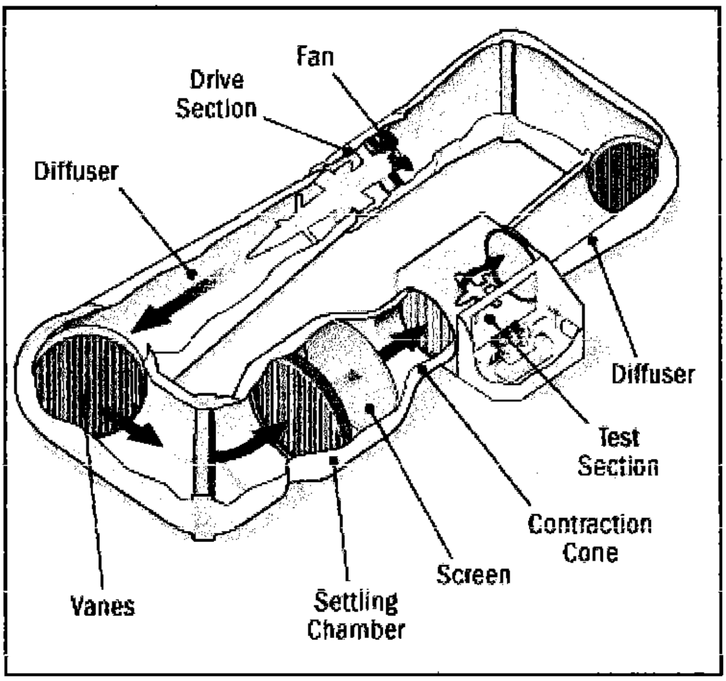


Figure 1.1: Model of a typical closed circuit wind tunnel
(www.science.howstuffworks.com/wind-tunnel3.htm)

Diffuser plays an important role of determining the performance of a wind tunnel. For closed circuit especially, the wind tunnels have oval shape like a race track, where the walls of the bending diffuser must exceedingly smooth to avoid any speed bumps that can cause turbulence. This will affect the evaluation and results upon the test of any object. Although diffusers are widely used, their flow characteristic is still not fully understood.

Whenever a uniform cross-section of a pipeline is interrupted by the inclusion of pipe fitting such as valve, junction, flow measurement device or bend, then a pressure loss will be incurred. The term "separation losses" will be chosen to define pressure losses across such fittings. Wide angle diffusers used in wind tunnel must be controlled with steady flow engineering device like mesh screen and honeycomb hole.

Generally, for the case of a sudden enlargement or diffuser, the flow separates from the pipe walls as it passes through the obstructing pipe fitting, resulting in the generation of eddies in the flow and consequently pressure loss occurs. As the bend becomes sharper, so the areas of separation become more extensive and the loss coefficient increases. Although diffuser angles larger than 5 degrees can increase the pressure recovery, it can also lead to boundary layer separation and unsteady flow.

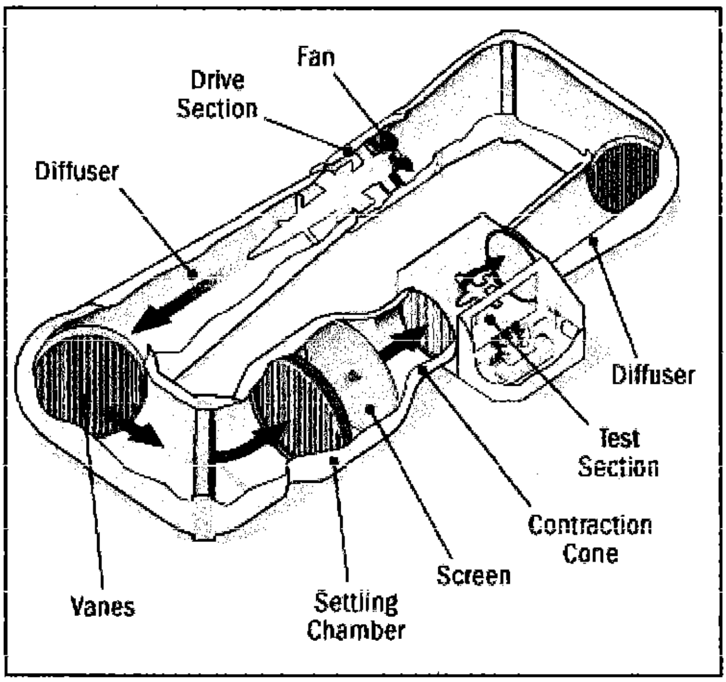


Figure 1.1: Model of a typical closed circuit wind tunnel
(www.science.howstuffworks.com/wind-tunnel3.htm)

Diffuser plays an important role of determining the performance of a wind tunnel. For closed circuit especially, the wind tunnels have oval shape like a race track, where the walls of the bending diffuser must exceedingly smooth to avoid any speed bumps that can cause turbulence. This will affect the evaluation and results upon the test of any object. Although diffusers are widely used, their flow characteristic is still not fully understood.

Whenever a uniform cross-section of a pipeline is interrupted by the inclusion of pipe fitting such as valve, junction, flow measurement device or bend, then a pressure loss will be incurred. The term "separation losses" will be chosen to define pressure losses across such fittings. Wide angle diffusers used in wind tunnel must be controlled with steady flow engineering device like mesh screen and honeycomb hole.

Generally, for the case of a sudden enlargement or diffuser, the flow separates from the pipe walls as it passes through the obstructing pipe fitting, resulting in the generation of eddies in the flow and consequently pressure loss occurs. As the bend becomes sharper, so the areas of separation become more extensive and the loss coefficient increases. Although diffuser angles larger than 5 degrees can increase the pressure recovery, it can also lead to boundary layer separation and unsteady flow.

Basically, the performance of a diffuser is evaluated in terms of its pressure recovery and flow uniformity. The maximum possible pressure recovery and flow uniformity can be obtained by setting the geometric and operating parameters optimally. As well as straight diffuser, turning diffuser could also produce losses and flow separation. Straight diffuser has slightly higher pressure recovery compared to turning diffuser. The turning and bending adopted in the turning diffuser could produce losses that could affect the performance of overall system and disrupt the uniformity of the flow.

Pressure loss reduction and flow uniformity can be achieved by introduction of baffles in the bending diffuser. Several studies were conducted to find the optimum location of baffles. Moore and Kline [1] has used five equally space flat vanes on a 45 degree wide-angle diffuser and resulting in separation free flow Other research by Majumdar et al. [2] shows improvements in the overall flow uniformity and performance for 90 degree curved diffuser.

1.1 Objectives

1. The objective of this study is to experimentally investigate the mechanism of flow structure and pressure recovery in turning diffusers by means of turning baffles.
2. It is also an attempt to propose an optimal configuration of baffles and evaluate the effectiveness of the new design to reduce pressure loss and increase flow uniformity.

1.2 Scope of study

1. A turning diffuser is here defined as a duct of tapered cross-section, connected to an elbow duct of uniform cross-section.
2. Baffles with improved geometries and arrangement will be installed.
3. Variables considered in this study are various Reynolds number.
4. The optimum relationships to be determined are on the pressure recovery coefficient and velocity distribution of the air flow which resembles flow uniformity.

1.3 Thesis structure

1.3.1 Problem statement

Diffusers form an important component of many fluid machines. Combined turning and diffusing flow contains much more complex flow and requires a lot of effort in measuring the overall performance. The turning or bending adopted in the ducting could produce losses that could affect the performance of overall system. The flow separation may occur which leads to pressure losses and disrupt the uniformity of the flow.

Efforts to overcome this problem have been done by adopting certain ratio of curvature to the bend, installing guided vanes, and varying the diffuser angle and spacer length. Most research resulting in improvement of flow uniformity and reducing pressure loss.

Uniform-area bends and elbows have a lot of unresolved research. At the same time, non-uniform area bends which are also call turning diffusers face the same problem. Since both designs are widely used in industrial applications, it is essential to investigate and understands the flow in turning diffusers and lead to the prediction of the pressure drop in them at varying Reynolds numbers.

Further modification can be done towards the previous baffle design and experimentally investigate the performance and effectiveness towards increasing pressure recovery and improve flow uniformity.

1.3.2 Significant of study

Since diffuser form an important component of many fluid machines, the investigations of the flow through diffusers are very important in understanding and improving their performances.

1.3.3 Expected outcomes

The performance of the turning diffuser in terms of flow structure and pressure recovery reduction by varying configuration of baffles in the events of different Reynolds Number could be determined. It is expected that the result for experiment of turning diffuser with the combination of new proposed design of turning baffles in the event of different Reynolds Numbers will offer better performance criteria.

CHAPTER 2

LITERATURE REVIEW

2.1 Flow separation and pressure variation

Fluid in motion has become the main concern for engineers. Flow in motion for liquid is somehow visible. But for gases, it's almost impossible to visualize the flow pattern. Eulerian and Lagrangian describe to identify a small mass of fluid named "fluid particle". To visualize the flow, a thin line can be constructed to show the flow direction from the fluid particle. These lines are called streamlines.

When a flow passes through around a body, the flow divided where half part of the streamline will go one way and another half will go the other way. At the location of where the flow divided are called stagnation point. The flow seems separated and the streamlines that did not follow the original streamline are the separation point as mention in Clayton et al. [3].

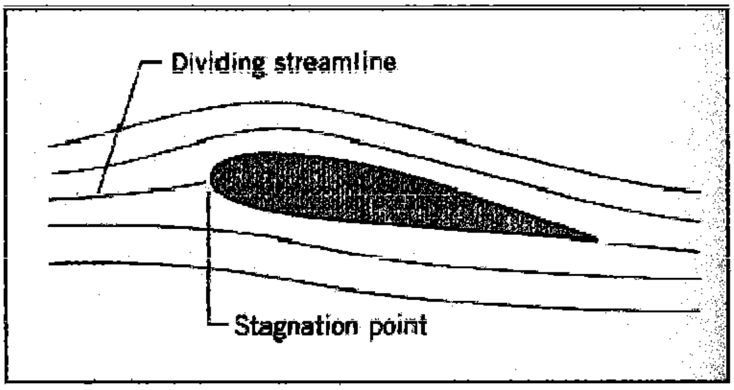


Figure 2.1: Flow separation over an airfoil shape (Clayton et al. [3])

Baffles helps to act like a barrier and prevent carryover of flow in inner wall towards the outer wall and helps flow uniformity as shown by Modi and Jayanti[4] in Figure 2.2 below.

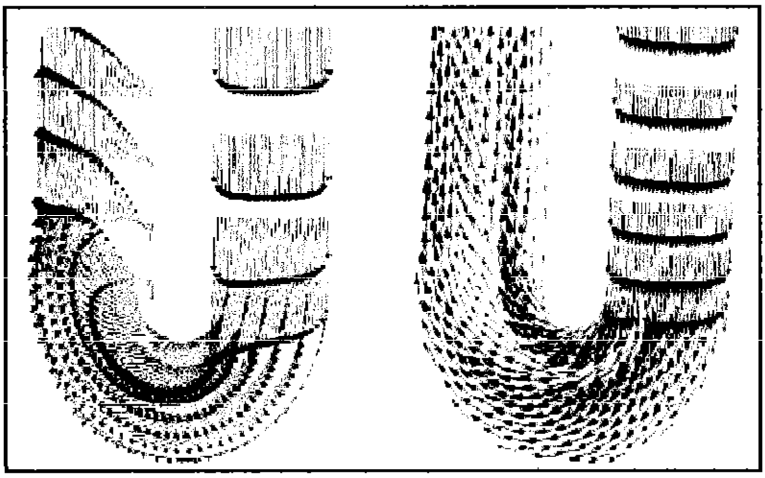


Figure 2.2: 180° bends with and without baffles (Modi and Jayanti[4])

Wake is form when flow separation occurs. The flow in the wake region is called separated flow. When the streamlines are joint back after pass through the body, there is no more separation occurs. This is due to form drag and consequently increases pressure losses.

There are other losses when a flow passes through a body which is called friction drag. Friction drag depends on the friction coefficient, C_D of a material of the body. Both drag and flow separation contributes to pressure losses in the turning diffuser and consequently reduce its performance.

The pressure difference at the outer and inner wall and flow maldistribution in the turning diffuser is due to flow separation and pressure mention above. To overcome these flaws, baffles are introduced. Baffles will act like a flow divider and the forces required to turn the streamline are distributed on a number of surfaces.

Turning diffuser without baffles will only have friction drag at both inner and outer wall only and flow separation at the inner wall. The effort of installing baffles contributes to additional friction drag at the surface baffles but improved the flow separations. This study will discuss further the effectiveness of turning diffuser with introduction of baffles.

2.2 Straight diffuser

Study regarding diffuser performance by mean of installing baffles started earlier since 1958 done by Moore and Kline [1]. Baffles in this term refer to a flow-directing vane or guide vane, a smart device for diffuser flow control. The authors started with the experiment on flow behavior in a subsonic diffuser by injecting dye in a water table to visualize the flow.

The variable for the first experiment are the divergence angle of the diffuser. Starting with 10° divergence angle resulting in no flow separation took place and varies the divergence angle up to 17° where the flow diverted by a separation of three-dimensional form. The beginning of flow separation was also marked as the beginning of an unsteady flow regime.

By means of installing baffle, they started with airfoil profile vane to accomplish the desired flow deflection. The vanes are necessarily located near the throat where the highest speed flow occurred. Other than this location is not suitable due to the dead-water region caused by flow separation and no kinetic energy to deflect. For this airfoil profile vane, the chord length used is 1 inch in length. The result from the airfoil profile vane, the separation still occurs. Because of this result, they change the design of the vanes to flat-plate vanes.

The flat vane used was made by galvanized iron. The metal was curved on the leading edge and slowly faired at the trailing edge. Vanes with 3 inches or shorter in length are 0.033 inch thick while vanes with longer length are 0.053 inch thick. For 20° divergence angle diffuser, a more stable separation than 17° divergence angle tested earlier occurs. To diminish the flow separation, they tried to locate a pair of 2 inch vanes at the downstream of the throat, but the separation was not entirely eliminated. They decided to change it to 3 inch in length for the pair of vanes and resulting in elimination of separation at all depth.

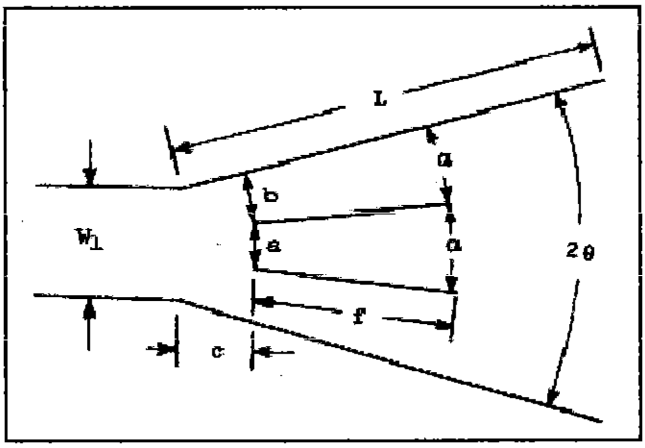


Figure 2.3: Typical vane design for straight diffuser (Moore and Kline [1])

Other than the dimension of the vanes, the location of each vane plays an important role towards their experiment. For the 30° divergence angle diffuser, with 3 guide vanes installed, the best solution is to set the vanes in equally spaced. This orientation will lead to 4 divergence angles of around 7.5° within the 30° divergence angle diffuser.

For the longitudinal placement, they had suggested the best location is about 1/4 to 1/2 inch downstream from the most narrow throat section. The downstream region is the region that has largest positive pressure gradients in the diffuser. The dynamic pressure and kinetic energy are also at their maximum level. High potential loss in diffuser performance at this region is the main reason of the need of goof throat flow configuration.

The pressure gradient at the throat is zero. It is no necessary to prevent separation from initiating at the throat. The slight divergence after the flow passes through the throat and before it reaches the leading edge of the vane is not sufficient to cause separation to occur.

From this experiment, they have outlined the criteria of a vane system as follows:

1. The optimum location for the vanes are slightly downstream from the narrowest diffuser throat
2. The number of vanes used should produce a small diffuser with maximum efficiency.

From this experiment, they have concluded that flat vanes can be very effective on preventing flow separation when located within a definite range of position. When vanes are equally spaced between one another and the divergence angle for each small diffuser gives maximum efficiency, maximum effect can be obtained from the flat vanes. For further increase of divergence angle of 45° with 5 equally space 6 inch vanes and located with leading edges almost at the throat has successfully remove all separations occurs in the diffuser.

For the cases of pressure recovery, in one of the test with 30° divergence angle diffuser, with addition of a pair of vanes, the pressure recovery factor increase up to 56% and efficiency increase up to 60%. In the other hand, with the increasing of flow rate in the same 30° divergence angle diffuser, with addition of a pair of vanes, the pressure recovery factor increase up to 11.5% and efficiency increase up to 13.5%. From this figure, they have proved that vane configuration improve the wide angle diffuser efficiency in terms of pressure recovery.

Moore and Kline [1] have investigated the starting behavior of flow in a diffuser by means of installing baffles. Many of their results and findings in this paper have been used by other researchers on improving diffuser performance. As for bending diffusers which are widely used in wind tunnel, the result might differ from what Moore and Kline [1] has concluded.

2.3 Turning diffuser

Lindgren et al. [5] started the studies for expanding bends, or so called turning diffuser or bending diffuser, both experimentally and numerically on the year 1998. In their experiment, the authors used diffuser with expansion ratios between 1 and 5/3 was tested. An open circuit wind tunnel was constructed for the special purpose to conduct this experiment. The variable in their experiment are area ratios, Reynolds number and vane spacing.

They mentioned in their experiment that secondary flow might occur over the vanes at low aspect ratio. To overcome this, the vanes are separated 8mm from the wall boundary layers. The vanes are made of aluminium extruded with 282mm span and 196mm chord and the aspect ratio of around 1.44.

The important parameters of designing the guide vane are guide vane chord, c, which refers to the distance from the leading edge to the trailing edge, the spacing between the guide vanes, d, and last but not least, the expansion ratio, e, which refers to the ratio of the distance between the guide vane at the trailing edge and the leading edge, respectively.

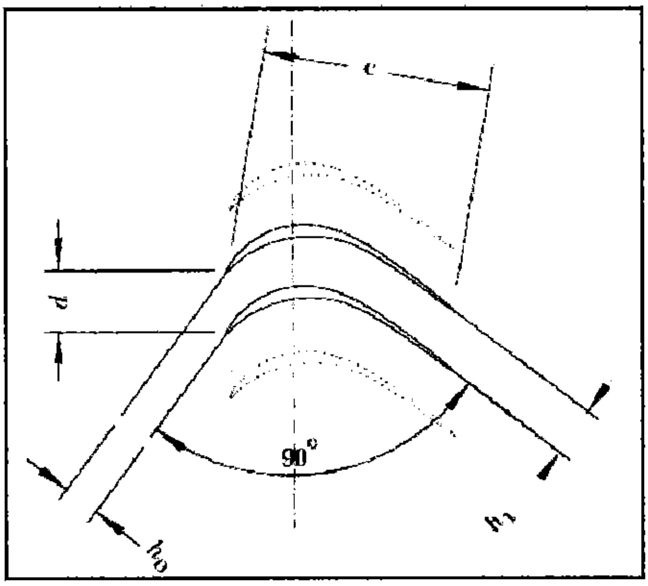


Figure 2.4: The geometry of an expanding cascade of guide vanes (Lindgren et al. [5])

They introduced the term pitch which resembles the spacing to chord ratio. The first measurement with pitch of 0.27 was made at expansion ratios 1, 4/3, 3/2 and 5/3. They have found that the total pressure loss coefficient increase when expansion ratio increase. A series of pitch ranging from 0.24 to 0.39 were conducted to find the optimum vane spacing which produces the lowest value of total pressure loss coefficient. The authors had found the optimum value of 0.35 for the value of the pitch. This design also results to total pressure loss coefficient of less than 0.054.

They performed the varying pitch with Reynolds number 200,000. For application with lower Reynolds number, separation tends to occur more easily thus the optimum value of the pitch will become lower than 0.35. They concluded that even though the design vanes are optimized for non-expanding corners, it can work well for expanding corners if the design can accommodate the lift coefficient.

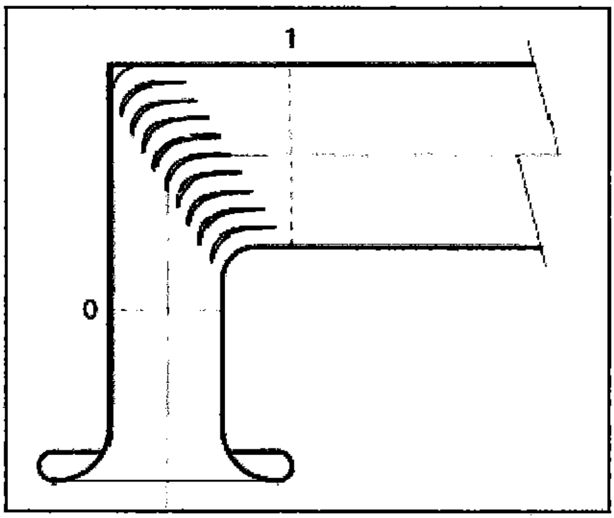
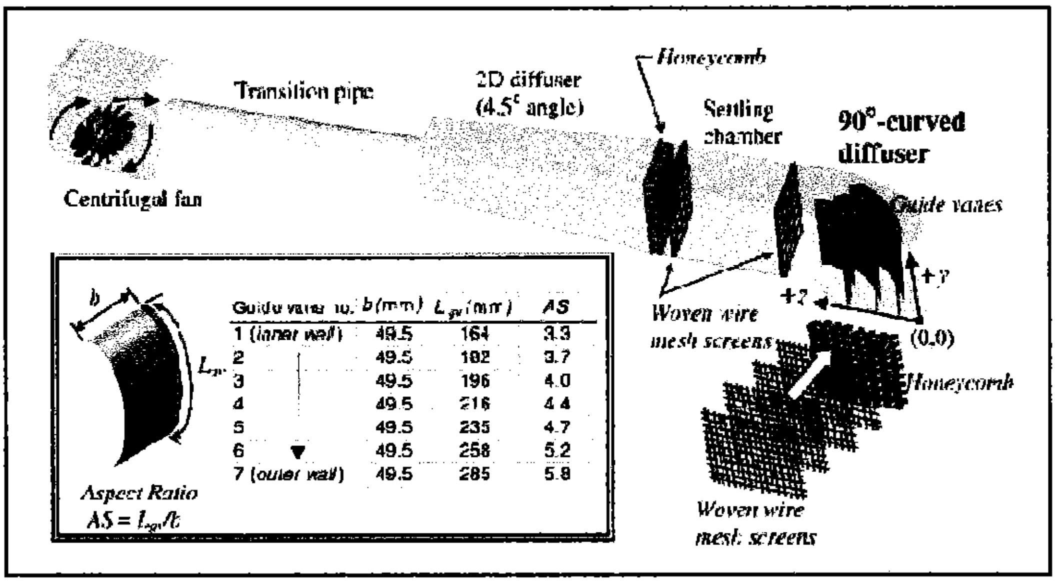


Figure 2.5: Guide vane configuration (Lindgren et.al [5])

After the experiment has successfully done, Lindgren [6] has introduced the idea of expanding corners in the construction of wind tunnels. But most experiments before this concluded that normal diffuser with non-expanding corners were a lot better. He concluded that the problem with previous experiments is poor design of guide vanes and must be optimized for each particular expansion ratio. It is likely that most guide vane can't sustain high loads and high pressure gradient present in the bending diffuser for wind tunnel purposes.

Chong et al. [7] has done an experiment to control the flow for a short, high area ratio 90° curved diffuser. They use several flow control device including vortex generators, woven wire mesh screens, honeycomb and guide vane inside the diffuser. Although the result was less successful for vortex generator, other device shows positive result when the cumulative pressure drop coefficient of 4.5 the least at the diffuser exit were achieved with minimum of 3 guide vanes to achieved flow uniformity.

In this experiment, the authors have tested up to 7 vanes with different profiles. They differentiate each vane according to their aspect ratio, AS. Aspect ratio is proportional relationship between width and height. The height of guide vane is the length of the curved section. Refer to Figure 2.6 for further understanding. The smallest AS will be located nearest to the inner wall and the configuration increasing toward the outer wall region.



Guide vane no.	b (mm)	L_{gv} (mm)	AS
1 (inner wall)	49.5	164	3.3
2	49.5	182	3.7
3	49.5	196	4.0
4	49.5	216	4.4
5	49.5	235	4.7
6	49.5	258	5.2
7 (outer wall)	49.5	285	5.8

Figure 2.6: Experimental facility and guide vane aspect ratio profile (Chong et al. [7])

In order to help reduce impact of boundary layer separation on the vane surface, they decided to use 0.8mm thin vane made of aluminium. Unlike Moore and Kline [1] did in their experiment for straight diffuser, Chong et al. [7] calculate the location of the guide vane. The leading edge of the guide vane was placed at the line in tangent with $L_{(i)}/W_1$=1.2. Referring to Figure 2.7, $L_{(i)}/W_1$ is the normalized distance from the curved-diffuser inlet at the inner wall. They believe that this configuration could minimize the boundary layer growth due to the reducing of AR of the passage between adjacent vanes.

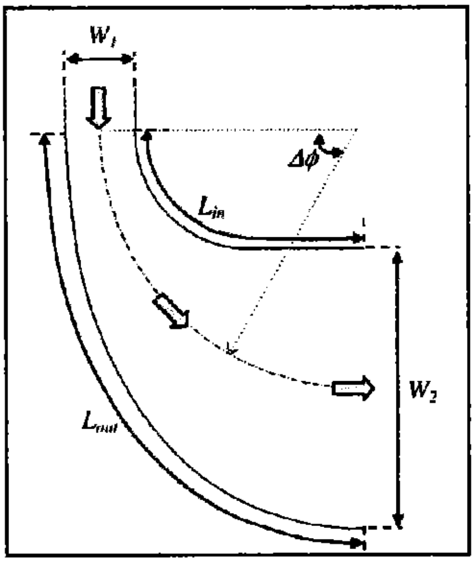


Figure 2.7: Parameters to described curved diffuser (Chong et al. [7])

A theoretical, pressure recovery coefficient denoted as C_p, distribution to the present 90° curved diffuser is shown in Figure 2.8 below. The C_p distribution is derived from potential flow theory and converting the present 90° curved diffuser into two-dimensional straight diffuser. As mention earlier, this experiment involves investigation upon other flow control device. The result are analyzed separately for each control device and compared to the base line C_p distribution.

The measured C_p for the base line case is calculated when no screen, guide vane and vortex generator (denoted as *q*, *gv*, and *vg* respectively) are install in the curved diffuser and represented with the notation (*q*=0, *gv*=0, *vg*=0).

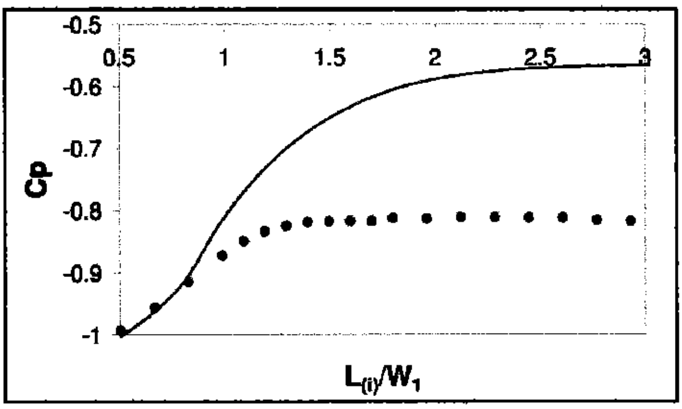


Figure 2.8: Comparison of theoretical (straight line) and measured C_p (dotted line) for the base line case (q=0, gv=0, vg=0) (Chong et al. [7])

Referring to the Figure 2.8 above, calculated C_p increase almost constantly from $L_{(i)}/W_1$ ranging from 0.5 to around 0.8 and further increase until it reach saturation at $L_{(i)}/W_1$ more than 2. For measured C_p, it increase almost the same like calculated C_p for $L_{(i)}/W_1$ ranging from 0.5 to around 1.19 but remain constant for the remaining $L_{(i)}/W_1$. This deviation shows that the flow boundary layer at the inner wall become unstable at $L_{(i)}/W_1 \approx 0.99$ and start to separate at $L_{(i)}/W_1 \approx 1.29$.

As for the result for installing guide vane, they had proved that significant impact can be seen in the pressure recovery if compared to the base line case. Figure 2.9 below shows that the flow separation can still be observed at $L_{(i)}/W_1 \approx 1.39$ indicated by the turning point where C_p distribution started to constant.

The figure shows clearly that with the addition of 7 evenly spaced guide vanes successfully improved the diffuser's exit flow uniformity but they might not be enough to completely eliminate the flow separation at the inner wall region. They supported this finding with findings by Majumdar et al. [8] where they still observed strong secondary flow even with the usage of high aspect ratio 90° curved diffuser.

It also shows a significant increase of pressure recovery as compared to the base line case. The C_p distribution patent is the same as the base line case but the value still higher as for the 7 evenly spaced guide vanes. Further experiments were conducted by varying the number of guide vane and other control flow device. Since the number of guide vane introduced only improve flow uniformity, with the constant pressure recovery improvement, the optimum 3 guide vane were concluded in this experiment.

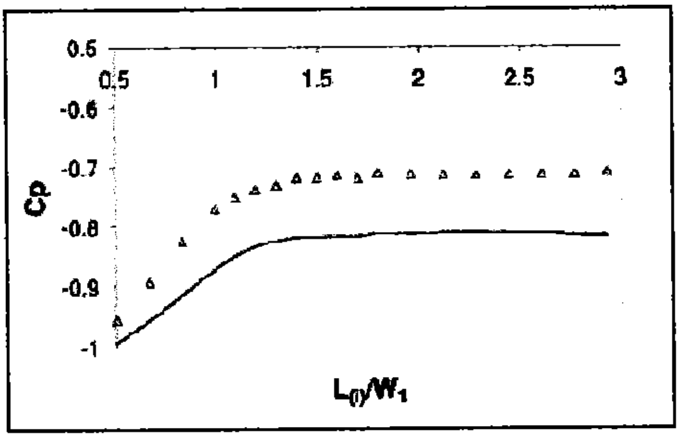


Figure 2.9: C_p distribution between base line (straight line) and q=0, gv=7, vg=0 case (Δ line) (Chong et al. [7])

Computational Fluid Dynamic (CFD) approach has been done by Eugene et al. [9] to compare the flow with and without the use of baffles in a bending diffuser. The authors use CFD with FLUENT code to simulate whether by means of baffles could increase pressure recovery and improvise flow uniformity.

2 units of baffles were installed with 0.2mm thickness and are not located at the beginning of inlet until end of outlet to avoid separation of flow to other section. The bending diffuser used has the area ratio of 2.2. First and foremost, they discuss the result of fluid flow in bending diffuser without baffles.

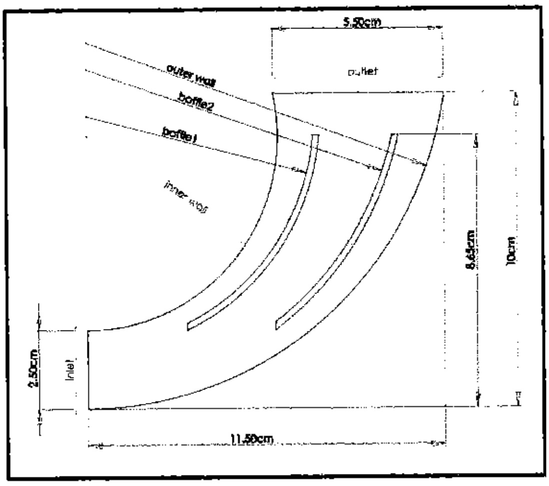


Figure 2.10: Turning diffuser with baffles installed (Eugene et al. [9])

It was shown that there is a flow separation concentrated at the outlet of the inner wall and causes the reduction of velocity to the flow. The flow separation was marked with the blue colour in Figure 2.11 below. When baffles are introduces, separation still occurs but with much less magnitude of separation. At the same time, little separation was seen at both leading and trailing edge of the baffle. The velocity at the outlet bending diffuser without baffles increases from outer wall to inner wall while the one with baffles were seen to have more symmetry velocity as shown in Figure 2.12 below.

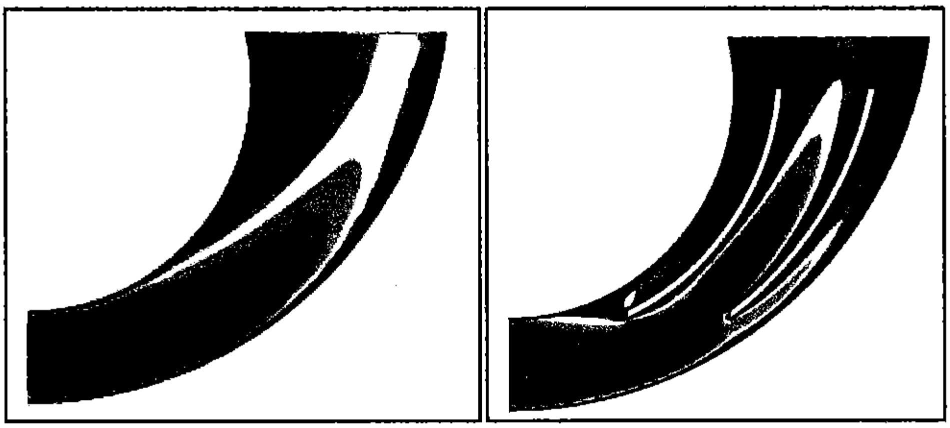


Figure 2.11: CFD result for flow without baffle (left) and with baffle (right) (Eugene et al. [9])

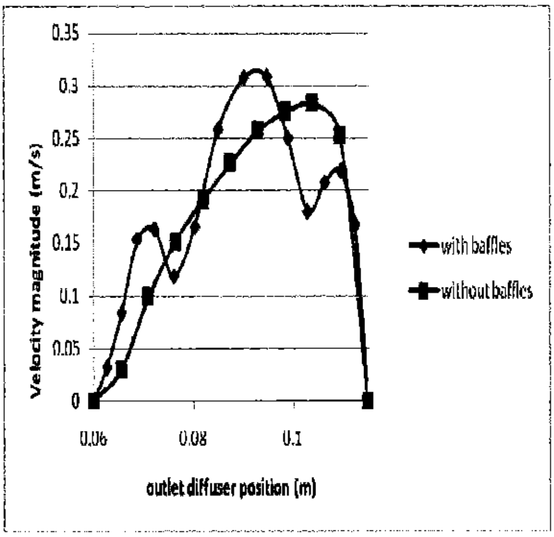


Figure 2.12: Velocity distribution at outlet bending diffuser (Eugene et al. [9])

The CFD result also shows the total pressure at the outlet turning diffuser is distributed evenly and total pressure at the inlet is much lower when introduced to baffle. As compared to turning diffuser flow without baffle, total pressure at the inlet is higher and decreases toward the inlet wall, marked by bright red yellowish colour in Figure 2.13 below. At the tip of the leading edge of the baffle also experienced high pressure due to form drag. They even proved this result with numerical approach.

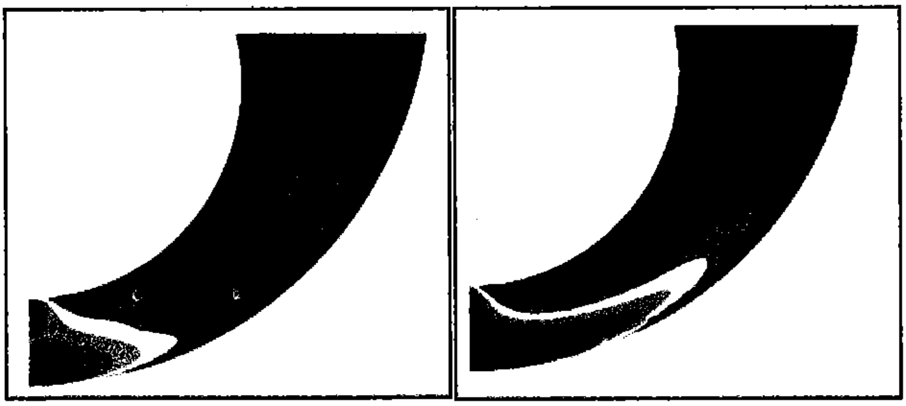


Figure 2.13: Total pressure in turning diffuser with and without installing baffles (Eugene et al. [9])

This effort is the starting point before continues with conducting the design in an experimental approach. Normayati et al. [10] continues the same investigation but with experimental approach. The type of vanes were ever tested in this experiment were guide vane design by Macbain [11].

On September 2012, Normayati et al. [12] conducted an experimental investigation of turning diffuser with an area ration of AR=2.16 operated at Reynolds number ranging from 5.786E+04 – 1.775E+05. To produce a prefect fully developed and uniform flow entering the diffuser, the experimental rig was introduced with several features of low subsonic wind tunnel.

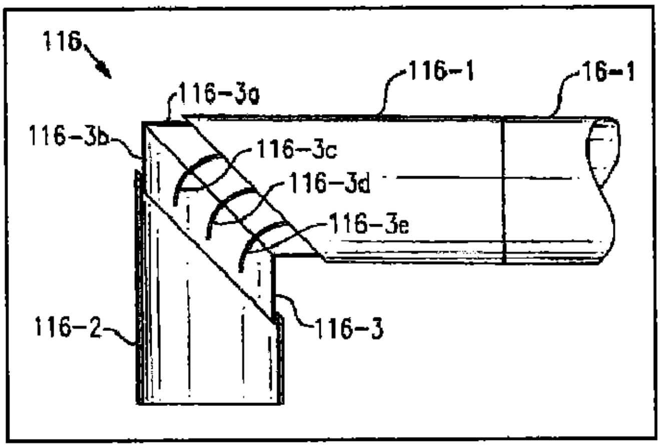


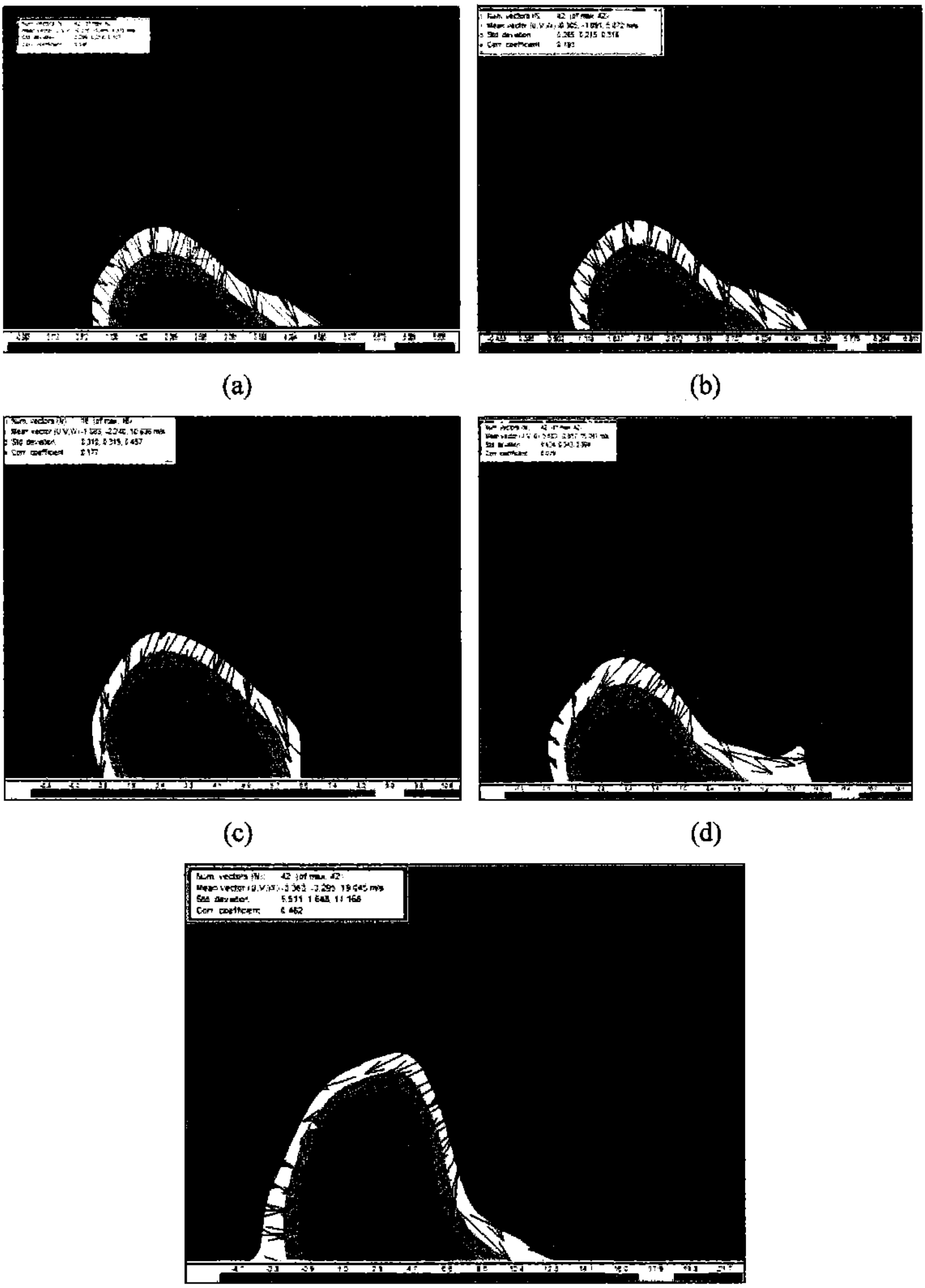
Figure 2.14: Design of guide vanes (Macbain [11])

To examine the flow uniformity, particle image velocimetry (PIV) was used whereas the average static pressure for both inlet and outlet turning diffuser where measured using digital manometer. The accuracy of the result from PIV were compared to manual measurement using Pitot static probe for both velocity at inlet and outlet diffuser. The best time between pulses is the least percentage of deviation between both.

According to Figure 2.15, Normayati et.al [12] shows the velocity measured using 3-D PIV was compared with the result using Pitot static tube probe at specific point. Table 2 shows the result of comparison between both methods.

Table 2.1: Verification of PIV results (Normayati et.al [12])

Re_{in}	$w_{pitot\ static}$	w_{piv}	Deviation (%)	The best Δt (μs)
5.786E+04	4.98	4.92	1.2	70
6.382E+04	5.92	5.87	0.8	70
1.027E+05	11.05	10.64	3.7	50
1.397E+05	15.45	15.34	0.7	30
1.775E+05	19.75	19.05	3.5	20



(a) (b) (c) (d) (e)

Figure 2.15: The third component velocity, w obtained by PIV was compared with the velocity measured using Pitot static probe at one particular point (a) Re_{in} = 5.786E+04 (b) Re_{in} =6.382E+04 (c) Re_{in}=1.027E+05 (d) Re_{in} = 1.397E+05 (e) Re_{in} =1.775E+05 (Normayati et.al [12])

Figure 2.16 illustrated the mean outlet velocity and standard deviation that resembles the flow uniformity of the outlet diffuser. For the flow structure within the diffuser, they have found that the boundary layer on the inner wall is likely to separate which leads to formation of pressure driven secondary flow. The inner wall boundary layer become thicker and exposed to flow separation as illustrated in Figure 2.17.

The result of this experiment also shows that C_p improves with increasing Re_{in} while flow uniformity is severely distorted, maximum up to σ_u=6.12. Table 2.2 summaries the result from the experiment.

Table 2.2: The effect of varying Re_{in} on C_p and K (Normayati et.al [12])

Re_{in}	C_p	K
5.786E+04	0.191	0.809
6.382E+04	0.209	0.791
1.027E+05	0.216	0.784
1.397E+05	0.221	0.779
1.775E+05	0.239	0.761

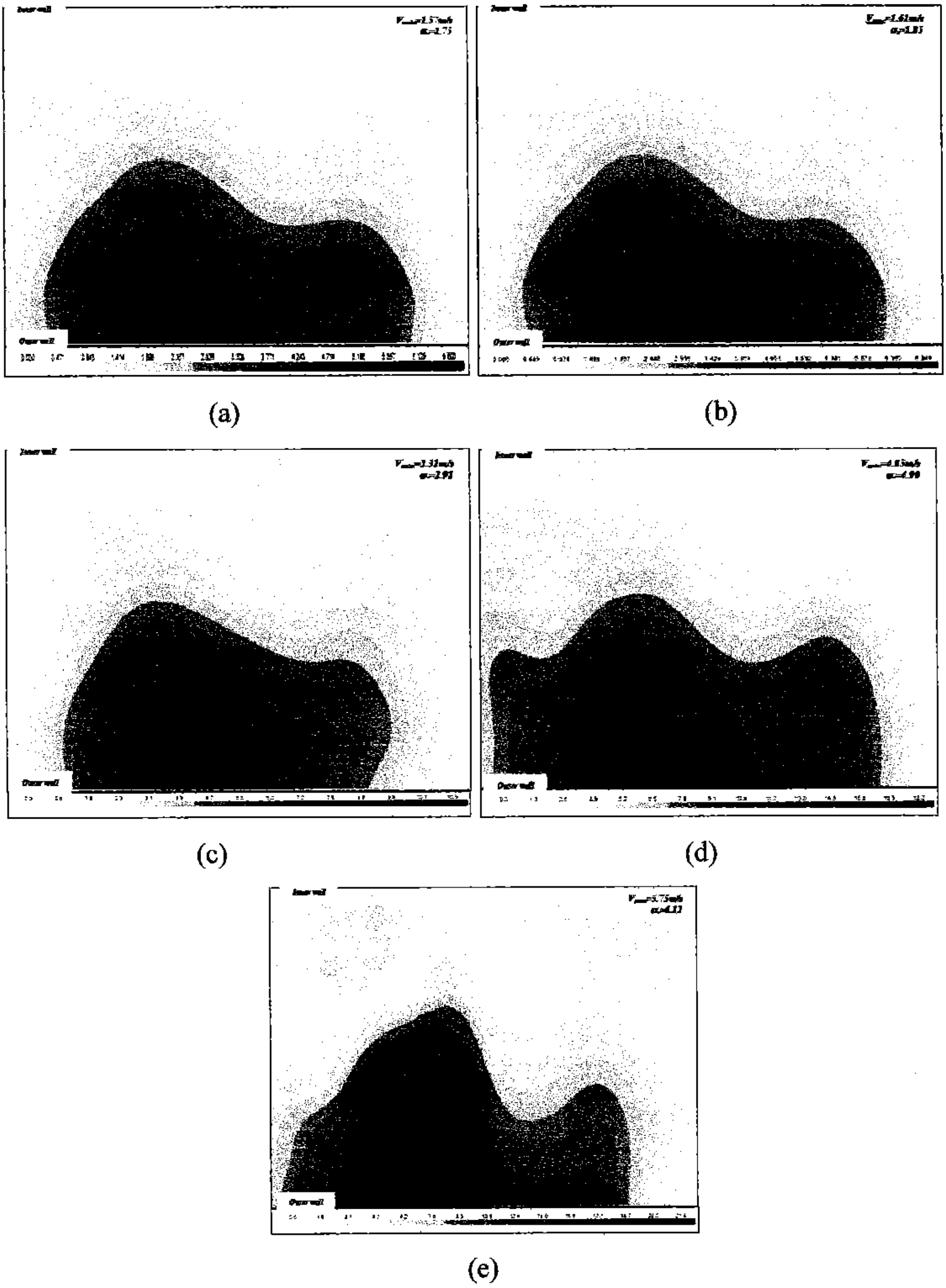


Figure 2.16: The outlet mean air velocity plane of turning diffuser operated at (a) Re_{in} = 5.786E+04 (b) Re_{in}=6.382E+04 (c) Re_{in} =1.027E+05 (d) Re_{in} = 1.397E+05 (e) Re_{in} =1.775E+05 (Normayati et.al [12])

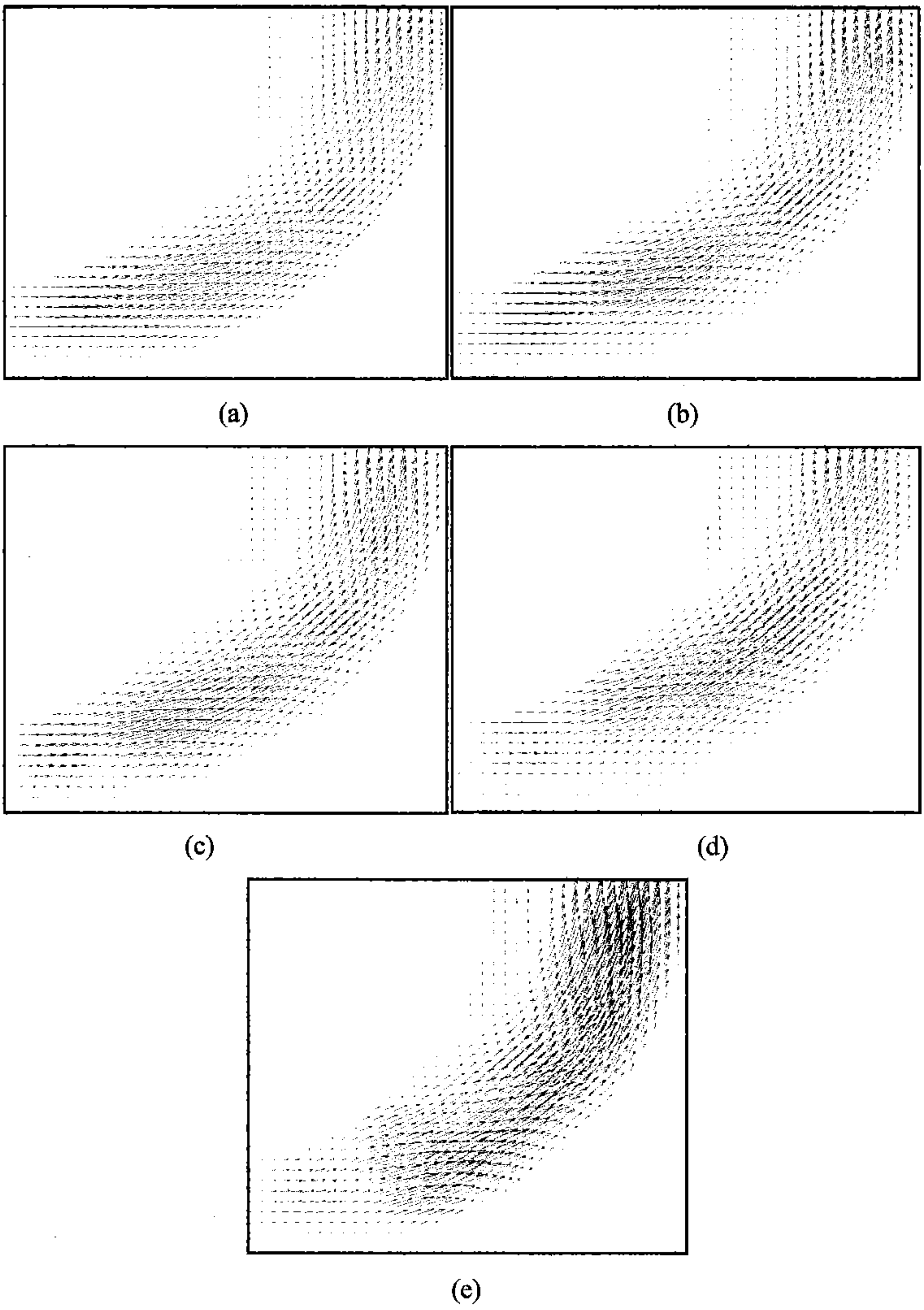


Figure 2.17: Flow structures within turning diffuser operated at (a) Rein= 5.786E+04 (b) Rein=6.382E+04 (c) Rein=1.027E+05 (d) Rein= 1.397E+05 (e) Rein=1.775E+05 (Normayati et.al [12])

REFERENCES

1. Moore, C. A., and Kline, S. J., "Some Effects of Vanes and of Turbulence in Two-Dimensional Wide Angle Subsonic Diffuser," NACA, Report No. TN-4080, 1958.
2. Majumdar, B., Singh, S. N., and Agrawal, D.P., "Flow Characteristics in a Large Area Ratio Curved Diffuser," J. Aerospace Eng., **210**(1), pp. 65-75, 1996.
3. Clayton T. C., Donald F.E., and John A.R., "*Engineering Fluid Mechanics*", Wiley International Edition, 8th ed., 2005.
4. Modi, P.P. and Jayanti, S., "Pressure Losses and Flow Maldistribution in Ducts with Sharp Bends, Chem. Eng. Res. Des. 82, pp.321-331, 2004.
5. Lindgren, B., Oesterlund, J., and Johansson, A. V., "Measurement Calculation of Guide Vane Performance in Expanding Bends for Wind Tunnels," Exp. Fluids, **24**(3), pp. 265-272, 1998.
6. Lindgren, B., and Johansson, A. V., "Design and Evaluation of a Low Speed Wind Tunnel With Expanding Corners," Royal Institute of Technology, Technical Report No. TRITA-MEK 2002:14, 2002.
7. Chong, T. P., Joseph, P. F., and Davies P.O.A.L., "A parametric Study of Passive Flow Control for a Short, High Area Ratio 90 Degree Cruved Diffuser," J. Fluids Eng., vol.130, 2008.
8. Majumdar, B., Mohan, R., Singh, S. N., and Agrawal, D. P., "Experimental Study of Flow in a High Aspect Ratio 90 Degree Curved Diffuser," ASME J. Fluids Eng., **120**(1), pp. 83-89, 1998.
9. Eugene, L. Z. P., N. Nordin, and S. Othman, "Numerical Investigation of Flow Maldistribution and Pressure Losses in Turning Diffusers by Means of

Installing Baffles," Conference Paper, Faculty of Mechanical Engineering and Manufacturing, University Tun Hussein Onn Malaysia (UTHM), 2010.

10. N. Nordin, S. Othman, V.R. Raghavan, S.M. Idris and M.F. Mohideen Batcha, "Experimental Investigation of Pressure Losses and Flow Characteristics in Bend-Diffusers by Means of Installing Turning Baffles," Conference Paper, Faculty of Mechanical Engineering and Manufacturing, University Tun Hussein Onn Malaysia (UTHM), 2011.

11. MacBain, S.M Chiller Compressor Circuit Containing Turning Vanes. United States Patent. Patent No. US 6,668,580 B2, 2003.

12. N. Nordin, Z.A.A. Karim, S. Othman and V. R. Raghavan, "Experimental Investigation of Turning Diffuser Performance by Varying Inflow Reynolds Number" PhD Progress Report for May 2012 Semester, Faculty of Mechanical Engineering and Manufacturing, University Tun Hussein Onn Malaysia (UTHM), 2012.

13. A. Sahlin and A. V. Johansson, "Design of Guide Vanes for Minimizing the Pressure Loss in Sharp Bends," Royal Institute of Technology, 1991.

14. Murthy, L. and Jie, C. "Numerical Investigation of Pressure Loss Reduction in a Power Plant Stack," Applied Mathematical Modelling 31, pp. 1915-1933, 2007.

15. J. Kim, P. Moin and R. Moser, "Turbulence statistics in fully developed channel flow at low Reynolds number", J. Fluid Mech., vol. 177, pp. 133-136, 1987.

16. DynamicStudio v3.12 User's Guide